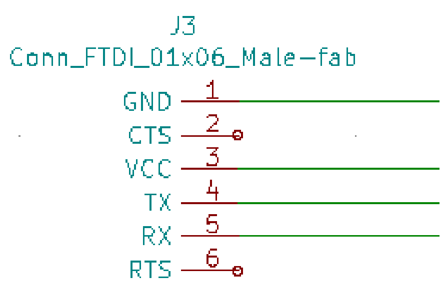
Associating Footprints with Symbols in KiCAD

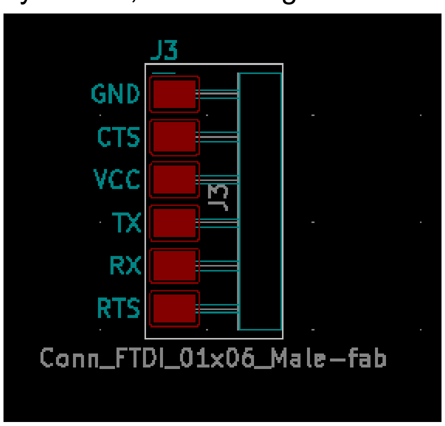
In KiCAD, the symbols in the schematic editor “Eeschema” can correspond to a variety of physical manifestations on your finished PCB. For example a resistor could be either a through-hole device or any of a variety of different surface-mount devices in different sizes. These have different “footprints” -- the pattern of holes and solder pads on the finished PCB.

In this week’s assignment, you might want to use “through-hole” header pins for your FTDI and UPDI connectors. These are stronger, and we currently have lots of them, whereas our stock of surface-mount headers is more limited.

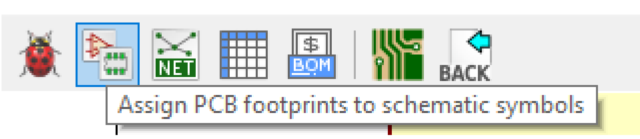
Here’s how to change the footprint for a symbol. We’ll start with a 6-pin FTDI connector:



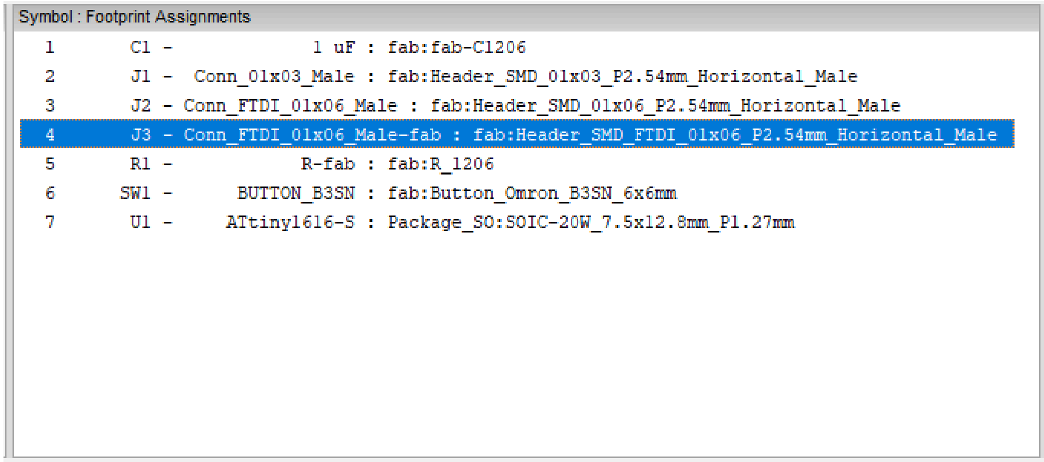
By default, this is assigned to a surface-mount footprint (square pads, no holes):



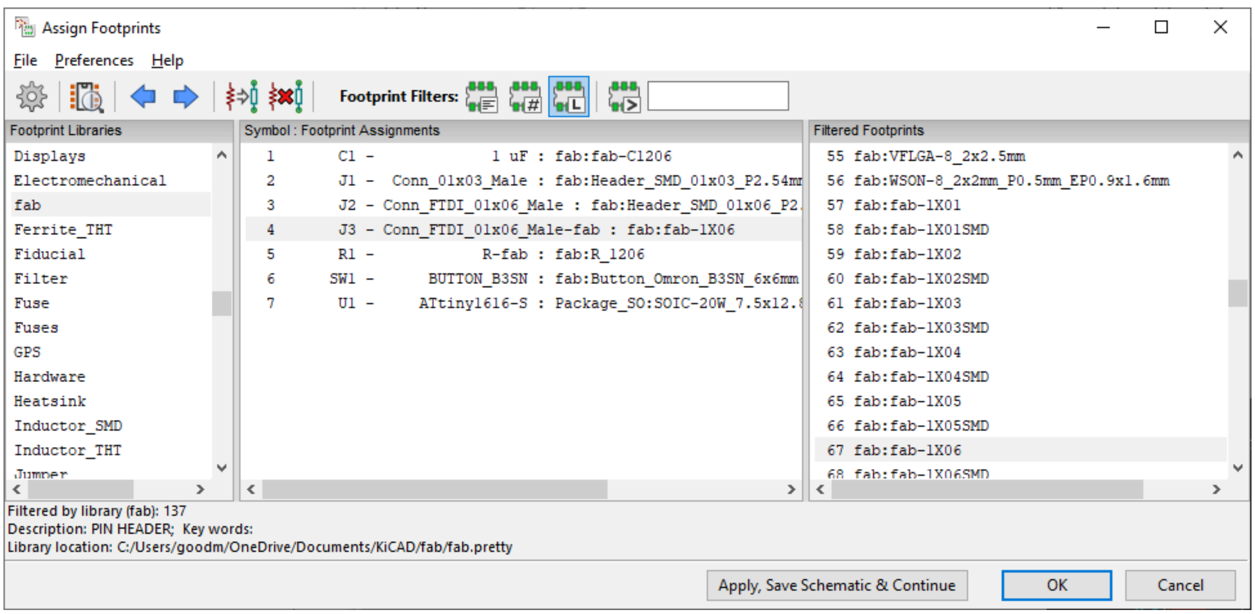
But we can change that using the “**Assign PCB footprints to schematic symbols**” button:



The center column shows each of the symbols, and the footprint it’s associated with:



Click the component you want to change in the center window, then choose the footprint library on the left column that has the desired footprint, and then **double-click** the specific footprint you want to use on the right column. Here, I'm using the Fab Academy library, which includes a 1x6 through-hole footprint. Notice how the center column changes to show the new footprint.



You can look at a footprint to see if it's what you want by right-clicking it and choosing "**view footprint**".

Then just click "OK".

The changes you make won't automatically affect your PCB: you'll have to "**update PCB from schematic**" in PCBnew.